Introduction to Abaqus

2016
Course objectives
Upon completion of this course you will be able to:

- Use Abaqus/CAE to create complete finite element models.
- Use Abaqus/CAE to submit and monitor analysis jobs.
- Use Abaqus/CAE to view and evaluate simulation results.
- Solve structural analysis problems using Abaqus/Standard and Abaqus/Explicit, including the effects of material nonlinearity, large deformation and contact.

Targeted audience
Simulation Analysts

Prerequisites
None
Day 1

Lesson 1  Overview of Abaqus

Demo 1  A First Look at Abaqus

Workshop 1  Linear Static Analysis of a Cantilever Beam

Lesson 2  Working with Geometry (Part 1)

Demo 2a  Working with Native Geometry

Demo 2b  Generating a Shell from a Solid

Workshop 2a  Creating Native Geometry: Pipe Creep Model

Workshop 2b  Creating Native Geometry: Reinforced Panel Model (Optional)

Lesson 3  Working with Geometry (Part 2)

Demo 3a  Generating a Shell From a Thin Solid

Demo 3b  Importing and Editing an Orphan Mesh

Workshop 3a  Import and Geometry Repair of Intersecting Pipes

Workshop 3b  Importing and Editing an Orphan Mesh: Pump Model
<table>
<thead>
<tr>
<th>Day 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lesson 4</td>
</tr>
<tr>
<td>Demo 4</td>
</tr>
<tr>
<td>Workshop 4a</td>
</tr>
<tr>
<td>Workshop 4b</td>
</tr>
<tr>
<td>Workshop 4c</td>
</tr>
<tr>
<td>Lesson 5</td>
</tr>
<tr>
<td>Demo 5</td>
</tr>
<tr>
<td>Workshop 5a</td>
</tr>
<tr>
<td>Workshop 5b</td>
</tr>
<tr>
<td>Lesson 6</td>
</tr>
<tr>
<td>Demo 6a</td>
</tr>
<tr>
<td>Demo 6b</td>
</tr>
<tr>
<td>Workshop 6a</td>
</tr>
<tr>
<td>Workshop 6b</td>
</tr>
<tr>
<td>Workshop 6c</td>
</tr>
<tr>
<td>Lesson 7</td>
</tr>
<tr>
<td>----------</td>
</tr>
<tr>
<td>Demo 7a</td>
</tr>
<tr>
<td>Demo 7b</td>
</tr>
<tr>
<td>Workshop 7a</td>
</tr>
<tr>
<td>Workshop 7b</td>
</tr>
<tr>
<td>Workshop 7c</td>
</tr>
<tr>
<td>Workshop 7d</td>
</tr>
</tbody>
</table>
Lesson 8
Demo 8a Using the Keywords Editor
Demo 8b Visualizing Results
Workshop 8a Creep of a Pipe Intersection
Workshop 8b Analysis of a Reinforced Panel (Optional)

Lesson 9
Demo 9 Multiple Load Cases
Workshop 9a Linear Static Analysis of a Cantilever Beam: Multiple Load Cases
Workshop 9b Linear Analysis of a Skew Plate

Lesson 10
Demo 10 Nonlinear Static Analysis
Workshop 10 Nonlinear Analysis of a Skew Plate
<table>
<thead>
<tr>
<th>Day 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lesson 11</td>
</tr>
<tr>
<td>Workshop 11</td>
</tr>
<tr>
<td>Lesson 12</td>
</tr>
<tr>
<td>Demo 12</td>
</tr>
<tr>
<td>Workshop 12</td>
</tr>
<tr>
<td>Lesson 13</td>
</tr>
<tr>
<td>Demo 13</td>
</tr>
<tr>
<td>Workshop 13</td>
</tr>
<tr>
<td>Lesson 14</td>
</tr>
<tr>
<td>-----------</td>
</tr>
<tr>
<td>Demo 14</td>
</tr>
<tr>
<td>Workshop 14a</td>
</tr>
<tr>
<td>Workshop 14b</td>
</tr>
<tr>
<td>Lesson 15</td>
</tr>
<tr>
<td>Demo 15</td>
</tr>
<tr>
<td>Workshop 15</td>
</tr>
<tr>
<td>Lesson 16</td>
</tr>
<tr>
<td>Demo 16</td>
</tr>
<tr>
<td>Workshop 16</td>
</tr>
</tbody>
</table>
## Additional Material

<table>
<thead>
<tr>
<th>Appendix</th>
<th>Title</th>
</tr>
</thead>
<tbody>
<tr>
<td>Appendix 1</td>
<td>Introduction to Finite Element Analysis</td>
</tr>
<tr>
<td>Appendix 2</td>
<td>Element Selection Criteria</td>
</tr>
<tr>
<td>Appendix 3</td>
<td>More on Contact</td>
</tr>
</tbody>
</table>
SIMULIA is the Dassault Systèmes brand for Realistic Simulation solutions

- Portfolio of established, best-in-class products
  - Abaqus, Isight, Tosca, fe-safe
  - All using a common extended licensing pool
SIMULIA’s Power of the Portfolio

Abaqus
- Routine and Advanced Simulation
- Linear and Nonlinear, Static and Dynamic
- Fluid, Thermal, Electrical, Acoustics
- Extended Physics through Co-simulation
- Model Preparation and Visualization

Isight
- Process Integration
- Design Optimization
- Parametric Optimization
- Six Sigma and Design of Experiments

Tosca
- Non-Parametric Optimization
- Structural and Fluid Flow Optimization
- Topology, Sizing, Shape, Bead Optimization

fe-safe
- Durability Simulation
- Low Cycle and High Cycle Fatigue
- Weld, High Temperature, Non-metallics

Realistic Human Simulation
High Speed Crash & Impact
Noise & Vibration

Material Calibration
Workflow Automation
Design Exploration

Conceptual/Detailed Design
Weight, Stiffness, Stress
Pressure Loss Reduction

Safety Factors
Creep-Fatigue Interaction
Weld Fatigue
Join the Community!

How can you maximize the robust technology of the SIMULIA Portfolio?
Connect with peers to share knowledge and get technical insights

Go to www.3ds.com/slc to log in or join!

Let the SIMULIA Learning Community be Your Portal to 21st Century Innovation
Discover new ways to explore how to leverage realistic simulation to drive product innovation. Join the thousands of Abaqus and Isight users who are already gaining valuable knowledge from the SIMULIA Learning Community.

For more information and registration, visit 3ds.com/simulia-learning.
SIMULIA SERVICES
PROVIDING HIGH QUALITY SIMULATION AND TRAINING SERVICES TO ENABLE OUR CUSTOMERS TO BE MORE PRODUCTIVE AND COMPETITIVE.

Training Schedule & Registration
We offer regularly scheduled public seminars as well as training courses at customer sites. An extensive range of courses are available, ranging from basic introductions to advanced courses that cover specific analysis topics and applications. On-site courses can be customized to focus on topics of particular interest to the customer, based on the customer’s prior specification. To view the worldwide course schedule and to register for a course, visit the links below.

North American
› By Location
› By Course

International
› By Location
› By Course

Live Online Training
› Full Schedule
The software described in this documentation is available only under license from Dassault Systèmes or its subsidiaries and may be used or reproduced only in accordance with the terms of such license.

This documentation and the software described in this documentation are subject to change without prior notice.

Dassault Systèmes and its subsidiaries shall not be responsible for the consequences of any errors or omissions that may appear in this documentation.

No part of this documentation may be reproduced or distributed in any form without prior written permission of Dassault Systèmes or its subsidiaries.

© Dassault Systèmes, 2015

Printed in the United States of America.

Abaqus, the 3DS logo, and SIMULIA are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries in the US and/or other countries.

Other company, product, and service names may be trademarks or service marks of their respective owners. For additional information concerning trademarks, copyrights, and licenses, see the Legal Notices in the Abaqus Installation and Licensing Guide.
<table>
<thead>
<tr>
<th>Lesson/Appendix</th>
<th>Revision Date</th>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lesson 1</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lesson 2</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lesson 3</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lesson 4</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lesson 5</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lesson 6</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lesson 7</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lesson 8</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lesson 9</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lesson 10</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lesson 11</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lesson 12</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lesson 13</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lesson 14</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lesson 15</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lesson 16</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Appendix 1</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Appendix 2</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Appendix 3</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
</tbody>
</table>

<p>| Demonstration 1 | 11/15 | Updated for Abaqus 2016 |
| Demonstration 2a | 11/15 | Updated for Abaqus 2016 |
| Demonstration 2b | 11/15 | Updated for Abaqus 2016 |
| Demonstration 3a | 11/15 | Updated for Abaqus 2016 |
| Demonstration 3b | 11/15 | Updated for Abaqus 2016 |
| Demonstration 4  | 11/15 | Updated for Abaqus 2016 |
| Demonstration 5  | 11/15 | Updated for Abaqus 2016 |
| Demonstration 6a | 11/15 | Updated for Abaqus 2016 |
| Demonstration 6b | 11/15 | Updated for Abaqus 2016 |
| Demonstration 7a | 11/15 | Updated for Abaqus 2016 |
| Demonstration 7b | 11/15 | Updated for Abaqus 2016 |
| Demonstration 8a | 11/15 | Updated for Abaqus 2016 |
| Demonstration 8b | 11/15 | Updated for Abaqus 2016 |
| Demonstration 9  | 11/15 | Updated for Abaqus 2016 |
| Demonstration 10 | 11/15 | Updated for Abaqus 2016 |
| Demonstration 11 | 11/15 | Updated for Abaqus 2016 |
| Demonstration 12 | 11/15 | Updated for Abaqus 2016 |
| Demonstration 13 | 11/15 | Updated for Abaqus 2016 |
| Demonstration 14 | 11/15 | Updated for Abaqus 2016 |
| Demonstration 15 | 11/15 | Updated for Abaqus 2016 |
| Demonstration 16 | 11/15 | Updated for Abaqus 2016 |</p>
<table>
<thead>
<tr>
<th>Workshop</th>
<th>Date</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Workshop 1</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 2a</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 2b</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 3a</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 3b</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 4a</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 4b</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 4c</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 5a</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 5b</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 6a</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 6b</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 6c</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 7a</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 7b</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 7c</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 7d</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 8a</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 8b</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 9a</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 9b</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 10</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 11</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 12</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 13</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 14a</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 14b</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 15</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 16</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
</tbody>
</table>
Lesson 1: Overview of Abaqus

Lesson content:

- What is Abaqus FEA?
- Abaqus/CAE
- Abaqus/Standard and Abaqus/Explicit
- Abaqus Conventions
- Working with the Model Tree
- Other Abaqus/CAE Topics
- Documentation
- Learning Community
- Abaqus Environment Settings
- Abaqus Fetch Utility
- Workshop Preliminaries
- Demonstration 1: A First Look at Abaqus/CAE
- Workshop 1: Linear Static Analysis of a Cantilever Beam

3 hours
Lesson 2: Working with Geometry (Part 1)

Lesson content:

- Abaqus/CAE: Part Module
- What are Parts?
- Creating Part Geometry
- Building a Part Using the Part Module Tools
- The Sketcher
- Adding Features
- Miscellaneous Topics
- Demonstration 2a: Working with Native Geometry
- Demonstration 2b: Generating a Shell from a Solid
- Workshop 2a: Creating Native Geometry: Pipe Creep Model
- Workshop 2b: Creating Native Geometry: Reinforced Panel Model (Optional)

2 hours
Lesson 3: Working with Geometry (Part 2)

Lesson content:

- Abaqus/CAE: Part Module
- Geometry Import and Repair
- Demonstration 3a: Generating a Shell From a Thin Solid
- Workshop 3a: Geometry Repair of Intersecting Pipes
- Part from an Orphan Mesh
- Creating Geometry from an Orphan Mesh
- Demonstration 3b: Importing and Editing an Orphan Mesh
- Workshop 3b: Importing and Editing an Orphan Mesh: Pump Model

1 hour
Lesson 4: Material and Section Properties

Lesson content:

- Abaqus/CAE: Property Module
- Abaqus Material Definitions
- Abaqus Conventions
- Linear Elasticity
- Large Strain Elasticity
- Metal Plasticity
- Material Calibration
- Material Databases
- Section Properties
- Special Features: Skins and Stringers
- Demonstration 4: Creating Materials and Assigning Sections
- Workshop 4a: Material and Section Properties: Pipe Model
- Workshop 4b: Material and Section Properties: Pump Model
- Workshop 4c: Material and Section Properties: Reinforced Panel Model (Optional)

2 hours
Lesson 5: Assemblies in Abaqus

Lesson content:

- Abaqus/CAE: Assembly Module
- What is an Assembly?
- Positioning Instances
- Other Operations
- Subassemblies
- Sets
- Surfaces
- Display Groups
- Instance Types
- Demonstration 5: Creating an Assembly; Boolean Operations
- Workshop 5a: Pump Model Assembly
- Workshop 5b: Reinforced Panel Model Assembly (Optional)

2 hours
Lesson content:

- Abaqus/CAE: Step Module
- Analysis Steps and Procedures
- Demonstration 6a: Creating Steps
- Output Requests
- Output Files
- Abaqus/CAE: Load Module
- Amplitudes and Distributions
- Loads and Boundary Conditions
- Initial Conditions
- Demonstration 6b: Using the Load Module
- Workshop 6a: Step Definition and Loads: Pipe Creep Model
- Workshop 6b: Step Definition and Loads: Pump Model
- Workshop 6c: Step Definition and Loads: Reinforced Panel Model (Optional)
Lesson content:

- Abaqus/CAE: Mesh Module
- What is a Mesh?
- Elements in Abaqus
- Mesh Generation Workflow
- The Mesh Module
- Common Tools:
  - Density
  - Controls
  - Element Selection
  - Meshing
  - Local Fine-tuning
  - Quality Checks
- Advanced Topics:
  - Virtual Topology
  - Bottom-up Meshing
  - Mesh Compatibility
  - Mesh Convergence
- Dependent and Independent Part Instances
- Demonstration 7a: Using the Mesh Module
- Demonstration 7b: Partitioning and Meshing
- Workshop 7a: Structured Hex Meshing: Pipe Creep Model
- Workshop 7b: Free and Swept Meshing: Pump Model
- Workshop 7c: Meshing of Intersecting Pipes
- Workshop 7d: Reinforced Panel: Meshing

Lesson 7: Meshing Imported and Native Geometry

2 hours
Lesson 8: Job Management and Results Visualization

Lesson content:

- Abaqus/CAE: Job Module
  - Analysis Jobs
  - Creating a Job
  - The Job Manager
  - Monitoring the Progress of an Analysis
  - Keywords Editor
  - Demonstration 8a: Using the Keywords Editor
- Viewing and Interpreting Results
- Abaqus/CAE: Visualization Module
  - Viewing and Interpreting Results
  - Output
  - Example 1: Overhead Hoist
- Example 2: Overhead Hoist – Dynamic Loading
- Example 3: Connecting Lug
- Additional Topics
  - Color Coding
  - Display Groups
  - Managing Viewports
  - Display Options
- Demonstration 8b: Visualizing Results
- Advanced Topics
  - Result Options
- Final Thoughts
- Workshop 8a: Creep of a Pipe Intersection
- Workshop 8b: Analysis of a Reinforced Panel

3 hours
Lesson content:

- Linear Static Analysis and Multiple Load Cases
- Multiple Load Case Usage
- Examples
- Demonstration 9: Multiple Load Cases
- Workshop 9a: Linear Static Analysis of a Cantilever Beam
- Workshop 9b: Linear Analysis of a Skew Plate
Lesson 10: Adding Material and Geometric Nonlinearity

Lesson content:

- Nonlinearity in Structural Mechanics
- Equations of Motion
- Nonlinear Analysis Using Implicit Methods
- Nonlinear Analysis Using Explicit Methods
- Including Nonlinear Effects in an Abaqus Simulation
- Diagnostics
- Convergence Issues (Noncontact Related)
- Demonstration 10: Nonlinear Static Analysis
- Workshop 10: Nonlinear Analysis of a Skew Plate

3 hours
Lesson 11: Multistep Analysis in Abaqus

Lesson content:

- Multistep Analyses
- Restart Analysis in Abaqus
- Workshop 11: Unloading Analysis of a Skew Plate

There is no Demonstration associated with this lesson.
Lesson 12: Constraints and Connections

Lesson content:

- Abaqus/CAE: Interaction Module
- Constraints vs. Connections
- Tie Constraints
- Rigid Bodies
- Coupling Constraints
- Shell-to-solid Coupling
- Connector Elements
- Fasteners
- Demonstration 12: Defining a Rigid Body
- Workshop 12: Tie Constraints: Pump Model

3 hours
Lesson 13: Contact

Lesson content:

- Abaqus/CAE: Interaction Module
- What is Contact?
- Approaches to Modeling Contact
- Defining General Contact
- Defining Contact Pairs
- Contact Pair Surfaces
- Local Surface Behavior
- Relative Sliding of Points in Contact
- Adjusting Initial Nodal Locations for Contact
- Contact Output
- For More on Contact
- Demonstration 13: Using Automatic Contact Detection and General Contact
- Workshop 13: Nonlinear Static Analysis of a Pump Assembly

3 hours
Lesson 14: Dynamic Analysis

Lesson content:

- What Makes a Problem Dynamic?
- Equations for Dynamic Problems
- Linear Dynamics
- Nonlinear Dynamics
- Comparing Abaqus/Standard and Abaqus/Explicit
- Nonlinear Dynamics Example
- Demonstration 14: Dynamic Analysis
- Workshop 14a: Dynamic Analysis of a Skew Plate
- Workshop 14b: Pipe Whip Analysis

3 hours
Lesson 15: Analyzing Nonlinear Quasi-Static Problems

Lesson content:

- Introduction
- Solution Strategies
- Quasi-Static Simulations Using Explicit Dynamics
- Adaptive Meshing
- Demonstration 15: Quasi-static Analysis
- Workshop 15: Single Pass Rolling of a Thick Plate

3 hours
Lesson 16: Heat Transfer and Thermal-Stress Analysis

Lesson content:

- Introduction
- Steady-State Heat Transfer
- Transient Heat Transfer
- Thermal Interfaces
- Thermal-Stress Analysis
- Demonstration 16: Thermal Analysis
- Workshop 16: Thermal-Stress Analysis of Intersecting Pipes

2 hours
Appendix 1: Introduction to Finite Element Analysis

Appendix content:

- What are Finite Elements?
- Objectives of Finite Element Analysis
- Components of an Analysis Model

0.5 hours
Appendix 2: Element Selection Criteria

Appendix content:

- Elements
- Structural (Shells and Beams) vs. Continuum Elements
- Modeling Bending Using Continuum Elements
- Stress Concentrations
- Contact
- Incompressible Materials
- Mesh Generation
- Solid Element Selection Summary
Appendix 3: More on Contact

*Appendix content:*

- Contact Issues Specific to Abaqus/Standard
- Contact Issues Specific to Abaqus/Explicit

1 hour